CFD MODELING AND STUDY OF COMBUSTION IN DI DIESEL ENGINE FOR DIFFERENT GRID GENERATION

[1]Vijayendra Patil, [2]Hemant Wagh, [3]Sachin Badgujar

[1]Assistant Professor, [2]Assistant Professor, [3]Assistant Professor
[1]Department of Mechanical Engineering,
[1]R.C. Patel Institute of Technology, Shirpur, India.

Abstract : Combustion is a chemical reaction in which certain elements of fuel like Hydrogen and Carbon combine with Oxygen liberating heat energy and causing and increase in temperature of the gases, within the engine cylinder. CFD approach can be a tool for gaining insight into the in-cylinder combustion process of diesel engine which is already a area of interest from recent years. It is the application of three dimensional codes, which are able to solve governing flow equations and yields detailed description of velocity, turbulence, pressure, temperature, etc. IC engines involve complex fluid dynamic interactions between air flow, fuel injection, moving geometries, and combustion. Fluid dynamics phenomena like jet formation, wall impingement with swirl and tumble, and turbulence production are critical for high efficiency engine performance and meeting emissions criteria. The design problems that are encountered include port-flow design, combustion chamber shape design, variable valve timing, injection and ignition timing, and design for low or idle speeds. In Internal combustion Engines CFD may be useful to study one of the most complicated process of combustion in Engines.

In this paper a CFD in-cylinder combustion analysis are carried out on a single cylinder four stroke Direct Injection (DI) Engine. The engine is 661cc Kirloskar engine. One of the main process in CFD Analysis is the Grid Generation. Proper and improve grids will give better and accurate results, Four grid cases having 327,1391,8413 and 35154 elements and 905,2830,15130 and 60797 nodes are considered for Grid Independence study. The optimum grid is find out of these four for CFD Combustion analysis and the computed data is validated with Experimental data available which are in good agreement.

Keywords - *: CFD, Engine Simulation, Grid Independence Study, Transient IC Engine Analysis.*

I. INTRODUCTION

High efficiency and least fuel consumption are always the areas of interest in IC Engine design. Generally, this were achieved by manufacturing prototype and conducting various tests on them. This process now can be said as a traditional process which is time consuming and cost consuming also this process is not versatile. If the prototype is used for single purpose it cannot be used for other experimentation purpose. Now a days, the field of computation is boosted by power of modern computers and workstations. CFD has found its applications in Diesel engine combustion. This method is widely used in industry not only for engine analysis but for a complete vehicle analysis.[4] CFD approach can be a tool for gaining insight into the in-cylinder combustion process of diesel engine which is already a area of interest from recent years. It is the application of three dimensional codes, which are able to solve governing flow equations and yields detailed description of velocity, turbulence, pressure, temperature, etc. Due to improvements in computational power and speed in modern computers CFD can be applied as it involves unsteady flows, high Reynold's number involved, high heat transfer and the complex variable geometry of engines. As a consequence, calculation times are usually long and computer memory requirement are large.[3]

Combustion is a chemical reaction in which certain elements of fuel like Hydrogen and carbon combine with oxygen liberating heat energy and causing an increase in temperature of the gases. The conditions necessary for combustion are the presence of combustible mixture and some means of initiating the process.[10] The important combustion parameters are Pressure, temperature and volume. These parameters affect the working of the engine and has a adverse effect on the performance of the engine. If these parameters are not in controlled manner or if they are not in certain defined limit they may cause engine problems such as Knocking which may cause severe damage to the vital components of engine. IC engines involve complex fluid dynamic interactions between air flow, fuel injection, moving geometries, and combustion. Fluid dynamics phenomena like jet formation, wall impingement with swirl and tumble, and turbulence production are critical for high efficiency engine performance and meeting emissions criteria. The design problems that are encountered include port-flow design, combustion chamber shape design, variable valve timing, injection and ignition timing, and design for low or idle speeds. [10] There are several tools which are used in practice during the design process. These include experimental investigation using test or flow bench setups, 1D codes, analytical models, empirical/historical data, and finally, computational fluid dynamics (CFD). Of these, CFD has the potential for providing detailed and useful information and insights that can be fed back into the design process. This is because in CFD analysis, the fundamental equations that describe fluid flow are being solved directly on a mesh that describes the 3D geometry, with sub-models for turbulence, fuel injection, chemistry, and combustion. Several techniques and sub-models are used for modeling moving geometry motion and its effect on fluid flow.

Using CFD results, the flow phenomena can be visualized on 3D geometry and analyzed numerically, providing tremendous insight into the complex interactions that occur inside the engine. This allows you to compare different designs and quantify the trade-offs such as soot vs NOx, swirl vs tumble and impact on turbulence production, combustion efficiency vs pollutant formation, which helps determine optimal designs. Hence CFD analysis is used extensively as part of the design process in automotive engineering, power generation, and transportation. With the rise of modern and inexpensive computing power and 3D CAD systems, it has become much easier for analysts to perform CFD analysis.[11]

CFD Approach plays a very vital role in Research and analysis from recent years. As this technique has become a part of every research field. In Internal combustion Engines CFD may be useful to study one of the most complicated process of combustion in Engines. By combustion analysis and simulation the plots of crank angle vs pressure, crank angle vs temperature and pressure vs

volume can be obtained which can be validate with the experimental results thus conforming the accurate of results obtained by using CFD. This method will be very advantageous to study process of combustion at various time steps during operation of engine by simulations thus reducing the experimental time and calculations. The high initial cost of experimental setup can be avoided as the results are obtained directly on computer screen within some time.

This method also has a flexibility to vary the engine geometry and operating conditions according to the user, which is not possible in experimental approach. Thus variations of these parameters will help in new processes in engine design and analysis. The actual operations taking place inside the engine cylinder can be visualized efficiently which is not possible for experiment situations or the experimental setup for the same will be costly.

II. CASE DESCRIPTION

The engine studied in this paper is a single cylinder direct injection (DI) diesel engine. The engine consists of one inlet valve and one exhaust valve with a centrally located fuel injector. CFD analysis of the combustion process in this engine is carried out by numerical method using a CFD package FLUENT. The main engine geometry consists of inlet manifold, exhaust manifold a fuel injector and piston bowl fig.1. During combustion in engines both the inlet and exhaust valves are closed and the combustion takes place above the piston usually known as combustion chamber[1]. Hence, in order to reduce the computational time a simplified model having 30° sector is consider for analysis. The model consists of a centrally located fuel injector and piston bowl which forms the combustion chamber fig.1.

Fig. 1 Complete model of Engine.

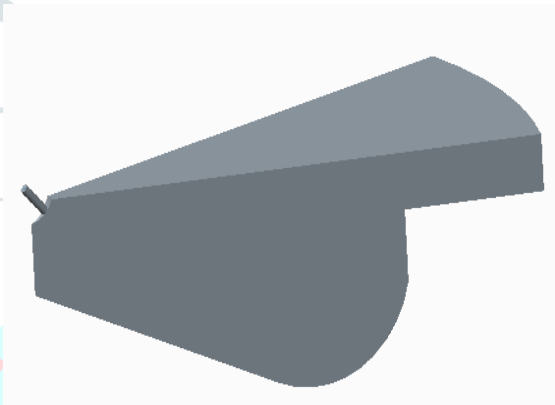

Fig. 2 A 30° sector model for analysis.

Various fluid zones and interfaces are provided for different regions of a model to enhance the results and for providing accurate interactions of boundary conditions. A dynamic mesh is carried out to simulate the suction and compression process up to the point of injection. After injection the monitors are applied for getting results of combustion parameters such as pressure, temperature and volume. The main aim of this paper is to find optimum mesh size by grid independence study which will give required results. Experimental study has been carried out on a Single cylinder four stroke DI Diesel engine with variable compression ratio. The numerical results will be compared for four mesh cases and optimum mesh case will be find out which will be in good agreement with experimental results.

Table 1shows the various engine parameters which are considered for this study.[10]

Parameter	Value
Cylinder Bore	87.5 mm
Connecting rod Length	234 mm
Stroke	110 mm
Engine Speed	1538 rpm
Compression Ratio	17.5
Inlet Valve open before TDC	4.5°
Inlet Valve close after BDC	36.5°
Fuel injection start before TDC	23°
Exhaust Valve open before BDC	35.5°
Exhaust Valve close after TDC	4.5°

Table 1 Engine Parameters.

III. NUMERICAL MODELING

In order to study the process of combustion a finite volume commercial flow program has been used to solve the discretized Navier Stokes equation. The standard *k-ε* turbulence model for high reynold's numbers with wall function is used.[3] The *k-ε* model for solving Navier stokes equation employees the eddy viscosity concept. Since k and ε both are produced destroyed and transported by flow, one obtains partial differential equations governing their values in the floe domain.[5] The code is based on pressure-

correction method using PISO algorithm. The first order upwind differencing scheme is used in the momentum and turbulence equations.[6] The governing equations for the gas flow and the combustion process are Mass equations, Momentum equations and Energy equations. Finite volume method has been used to solve the discretized partial differential equations governing the mean fluid motion. The discretization of space in the appearance of cell numbers and time in the form of time stepping are set which provide a reliable indication of the suitability of the discretization settings. The time discretization is necessary to obtain the best compromise between the convergence requirements and the computing time i.e. the smaller time steps were set for initiating simulations when the crank angle reached the fuel injection timing and the TDC position. [7]A species transport model with discrete phase has been used for non-premixed combustion with non-adiabatic equilibrium phase model. Injection is define at specific crank angle at which the fuel injection starts and stops. The fuel temperature, pressure and mass flow rate is defined during injection. The fuel injection parameters like location of injector, size of injector, injection temperature and pressure, mass flow rate etc. are having significant effect in diesel combustion modeling.[4] The droplet model of injection with injection location is specified. The surface monitor is defined to find Pressure, Temperature and volume from 20° before injection to end of combustion. As discuss earlier in Table 1 the fuel injection starts at 23° before TDC means at 337° CA and the combustion ends at 397° CA hence simulation starts at 317° CA and ends at 397° CA. Iso surface is created to plot contours of temperature and pressure for the combustion simulation. As explained earlier that the time discretization is necessary to obtain the best compromise between the convergence requirements so time step plays a very important role in solving combustion problems. Hence, time step of 0.25[2] is found optimum for combustion analysis. The volume monitors are obtained at various zones in the form of output data with respect to flowtime and time step. The number of iterations required for solving the combustion problems depends upon the time step and the CA to run. The computational time required for such simulations is more as they involved Dynamic mesh and Transient analysis.

IV. GRID INDEPENDENCE STUDY

A grid generation program has been used to create hexahedral grid. As explained earlier, due to complexity of defined problem a simplified model of 30° sector is used for analysis. The inlet port, exhaust port, Inlet and exhaust valve are ignored as these components does not have any significance role during combustion. Due to the fact that both the inlet and exhaust valves are closed during combustion and the fuel is injected before the end of combustion stroke i.e. at 23° before TDC which means at 337° CA. To study the process of combustion effectively simulation will be carried out some CA before fuel injection and after fuel injection. For this study the simulation will start at 317° and end at 390°.Generation of grid has a greater influence on solutions in CFD , a proper and better grid will provide accurate solutions. The Hexahedral cells have been adopted for mesh generation because they provide better accuracy and stability as compared to Tetrahedral cells.[8]The main purpose of using hexahedral element is its requirements of moving mesh. Dynamic mesh is carried out for this problem in order to study the In-cylinder process going on inside the engine cylinder. Hence, dynamic mesh starts at 0° CA up to 317° CA which is the starting point of our simulation, the number of time for dynamic mesh is calculated upon the number of timestep defined for analysis. A timestep size of 0.25 is found to be impressive for in-cylinder simulations which provide better accuracy and improve the results for dynamic mesh problems.[9]

Four mesh cases are considered for this problem Case I, Case II, Case III and Case IV. Table 2 shows the description for all the above Cases considered for Grid independence study for CFD Simulations.

Case	Face Sizing	No. of Nodes	No. of Elements
Case I	10 mm	905	327
Case II	5 mm	2830	1391
Case III	2 mm	15130	8413
Case IV	1 mm	60797	35154

Table 2 Description of Grid Cases.

The model is first modeled in 3D software with 3 bodies and 12 zones. The 3 bodies are fluid-inner, fluid-outer and fluid-inj which represents the inner and outer location of the combustion chamber from fuel injector. Several zones are defined for this two bodies which represents the fluid interaction and boundary conditions during simulation. Table 3 shows various fluid zones and their zone type.

Sr.no	Zone name	Zone type
1	Wall_top_inner	Stationary
2	Wall_top_outer	Stationary
3	Wall	Wall
4	Wall_inj	Wall
5	Period_inner1	Interface
6	Period_inner2	Interface
7	Period_outer1	Interface
8	Period_outer2	interface
9	Intf_inner	Periodic
10	Intf_outer	Periodic
11	inlet	Inlet
12	bowl	Rigid body

Table 3 Various Zones defined for analysis.

The zone names and bodies are specified before meshing. Providing zone names reduces the complicated work of providing boundary conditions for each surface of the model. After providing the zone names mesh is generated. the zones are further used

during solution for creating interface between periodic regions as we have modeled a 30° sector, When defining the zones and boundary conditions is completed the dynamic mesh is carried out to ensure that the cylinder top is stationary and the piston bowl moves in a required direction. The four Mesh cases considered for analysis is shown in fig.3 to fig. 6.

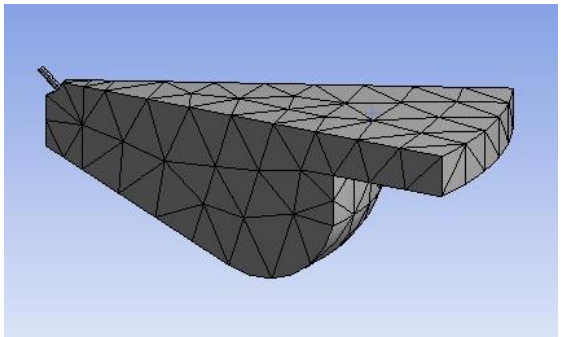

Fig. 3 Mesh Case I.

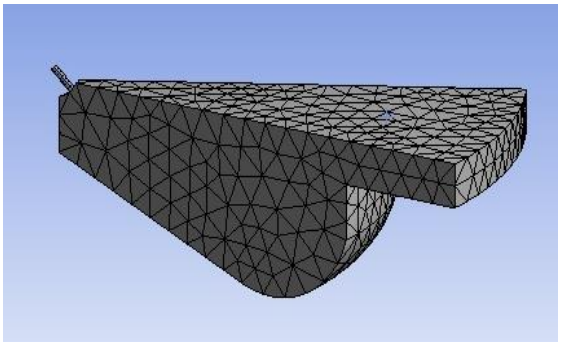

Fig. 4 Mesh Case II.

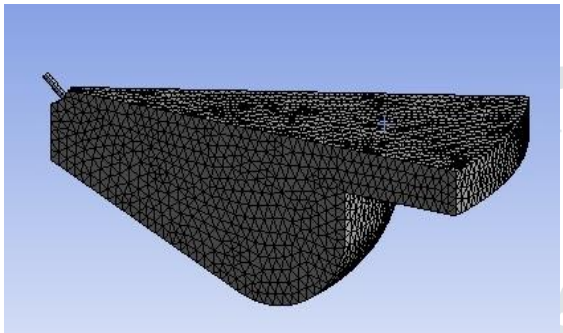

Fig. 5 Mesh Case III.

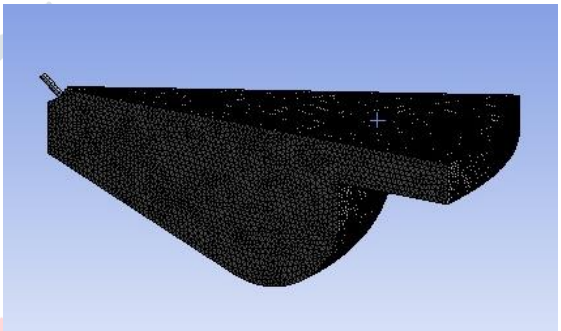

Fig. 6 Mesh Case IV

In order to ensure grid independence and find optimum grid and also to improve the accuracy of the results, Calculations were performed using Four Cases from which Case III and Case IV found results which were in good agreement with the experimental results. The experimental data is available for the same case hence, first the Pressure, Temperature and Volume is compared with all the four Cases shown in Fig. 7 to Fig. 9. The Case III is taken as the reference grid and the further calculations are performed using Case III. As Case III and Case IV provides same results but number of Nodes and Elements for Case IV are greater than Case III. so using Case III as reference grid will reduce the computational time and the probability of accuracy will increase.

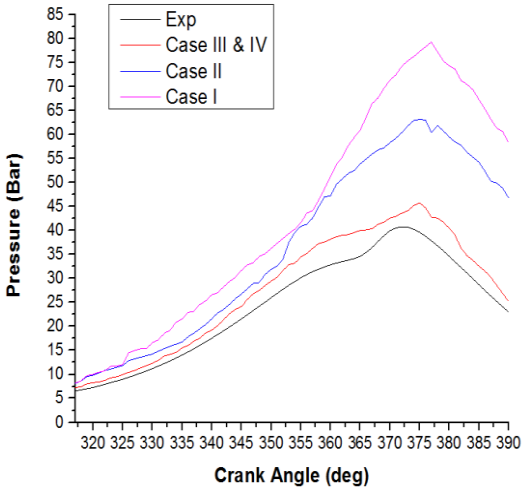

Fig.7 Pressure comparison for grid independence study.

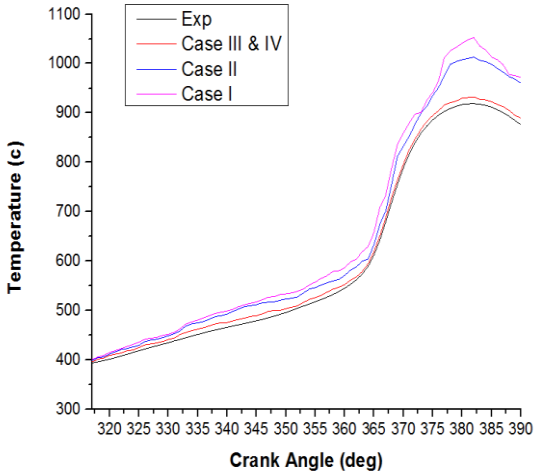

Fig.8 Temperature comparison for grid independence study.

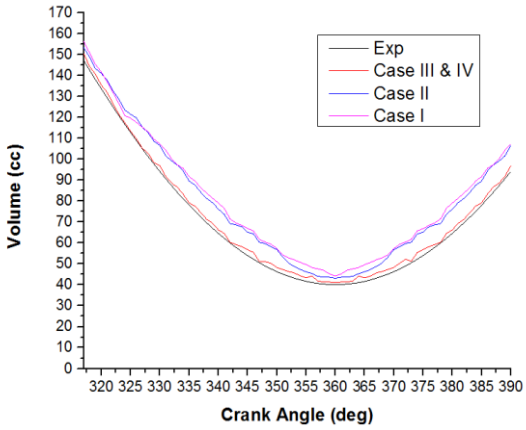

Fig. 9 Volume comparison for grid independence study.

From the above plots of Pressure (Bar), Temperature (c) in and Volume (cc) VS Crank Angle it is found that Case III& Case IV results are optimum.

V. RESULTS AND DISUCSSIONS

From Grid independence study the mesh with Case III has found better for results. The CFD analysis are carried out with Case III by using Numerical methods discussed in Numerical Modeling. The time step for analysis is 0.25 with 292 iterations. The simulation starts at 317° the fuel injection starts at 337° and simulation ends at 390°. The combustion process is studied from 317° CA to 390° CA in order to get insight on In-cylinder process and detail study of In-cylinder parameters such as Pressure, Temperature and Volume. The Experimental data obtained from VCR (Variable compression ratio Engine) is compared with CFD solution. The combustion process inside the engine cylinder can be simulated using contour plots of surface created for visualization of Temperature and Pressure at various Crank angles. The CFD simulation could give an idea of Physical Phenomenon occurring inside the engine cylinder which could not be possible in practice. Plots for various parameters such as Pressure, Temperature and volume are obtained in a tabular data format which can be plotted against Crank angle. The results are shown from Fig. 10 to Fig.15.

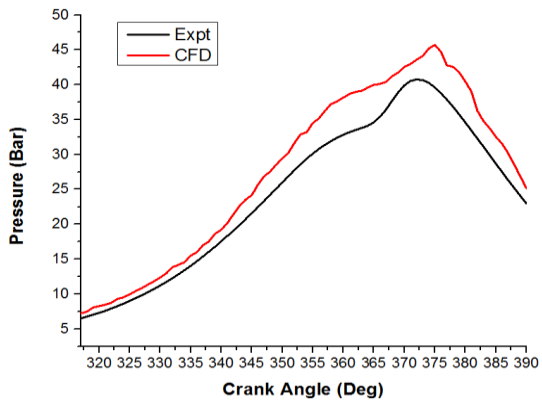

Fig. 10 Comparison of Experimental and CFD Pressure.

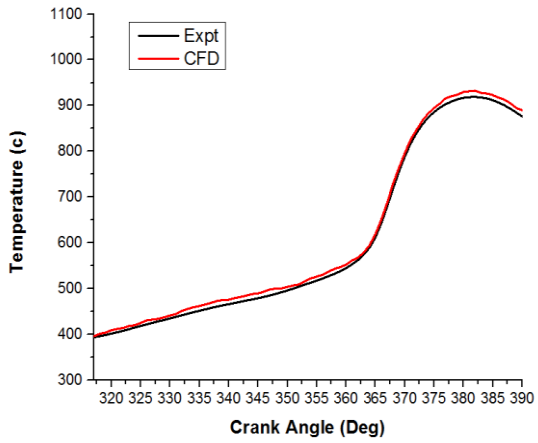

Fig. 11 Comparison of Experimental and CFD Temperature.

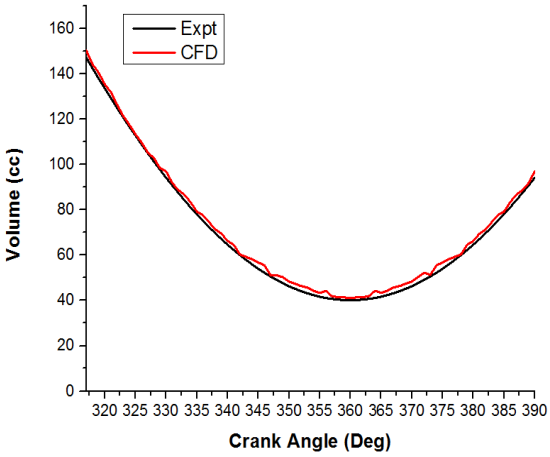

Fig. 12 Comparison of Experimental and CFD Volume.

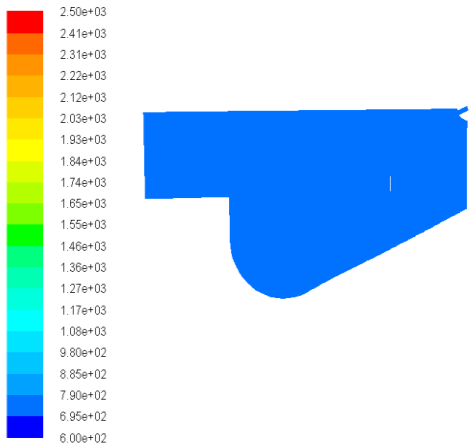

Fig.13 Contour of Temperature at 327° CA.

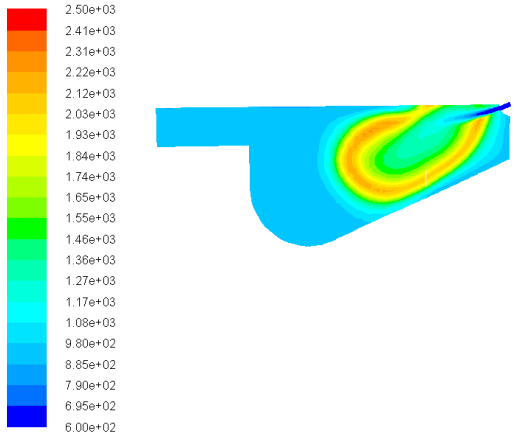

Fig.14 Contour of Temperature at 342° CA.

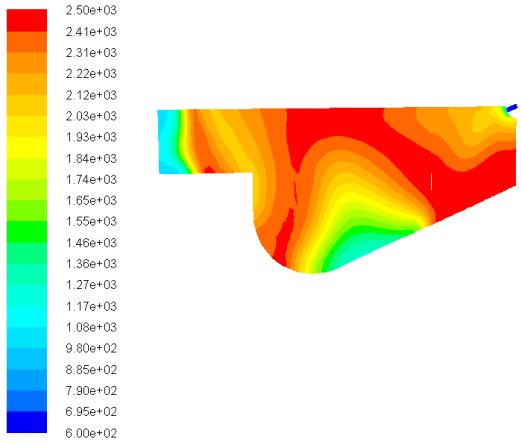

Fig.15 Contour of Temperature at 387° CA.

VI. CONCLUSION

A CFD combustion Simulation has been carried out for a Four stroke single cylinder DI engine. Due to complexity of problem only a 30° sector of a model is modeled in a 3D software. Model is defined with various Zones to define the boundary conditions and interfaces, Dynamic mesh is carried out In-cylinder simulation. Four cases of Grid are considered for Grid independence study, from which optimum Mesh which provides good results is selected. The various solver methods of a commercial CFD package is used for solving the problem.

Validation of Results is done with the good quality mesh results by comparing it with Experimental data obtained. The plots of Pressure, Temperature and Volume are plotted against Crank angle. The contours of Pressure at various Crank angles are visualized for simulation and insight to In-cylinder combustion processes which is not possible practically. The plots shows that CFD results are in good agreement with Experimental results with a maximum difference of 15%.
In general, this study shows that In-cylinder CFD simulation and study yield accurate results that allow improving the knowledge of combustion in DI diesel engines. The work presented in this paper insights the simulation process inside the engine cylinder, CFD has become modern tool to study the complicated process of combustion in engines, Instead of using highly cost experimental setup this technique can be adopted for combustion analysis of DI Diesel engines, which could play a very important role in engine design, research and development.

REFERENCES

[1]. Vijayendra Patil , Aashish Agrawal " In Cylinder Combustion Analysis of DI Diesel Using Experimental and CFD Approach." International Journal of Engineering Trends and Technology (IJETT) Vol. 4, Number 5, Aug 2014, pp.249-254.

[2]. Patil Vijayendra Maharu, Aashish Agrawal "Optimization of Time Step and CFD Study of Combustion in DI Diesel Engine." International Journal of Research in Engineering and Technology (IJRET), ISSN:2319-1163, Vol.03, Jun 2013 pp. 22-26.

[3].F. Payri , J. Benajes, X. Margot , A. Gil, "CFD modeling of the in-cylinder flow in direct-injection Diesel Engines." Computers & Fluids 33 (2004) 995–1021.

[4].S Gavudhama Karunanidhi, Melvinraj C R, Sarath Das K P, G Subha Rao, "CFD Studies of Combustion in Diesel Engine."International Journal of Engineering Research and Applications (IJERA),ISSN: 2248-9622,Vol. 3, Issue 4, Jul-Aug 2013, pp.827-830.

[5].S.M. Jameel Basha ,P. Issac Prasad and K. Rajagopal, "Simulation of In-Cylinder processes in a DI Diesel Engine with Various Injection Timings" ARPN Journal of Engineering and Applied Sciences, Vol, 4, NO:1, Feb 2004. ISSN 1819 - 6608.
[6].A.Hatziapostolou, G.Raptis, "CFD Modeling if In-Cylinder flow in a Variable Compression Spark-Ignited Engine "Proc. of the 9th WSEAS Int. Conf. on Mathematical and Computational Methods in Science and Engineering, Trinidad and Tobago, November 5-7, 2007.

[7]. Shahrir Abdullah, Wendy Hardyono Kurniawan and Azhari Shamsudeen, "Numerical Analysis of the Combustion Process in a Compressed Natural Gas Direct Injection Engine", Journal of Applied Fluid Mechanics, Vol. 1, No. 2, pp. 65-86, 2008.ISSN 1735-3645.

[8]..Wendy Hardyono Kurniawan, Sharir Abdulla and Azhari Shamsudeen, "A Computational Fluid Dynamics Study of Cold Flow Analysis For Mixture Preparation In a Motored Four-Stroke Direct Injection Engine "Journal of Applied Sciences 7 (19) : 2710-2724,2007. ISSN 1812-5654.

[9]. T. Morauszki, P. Mandli ,Z. Horvath and M.R .Dreyer, "Simulation of Fluid Flow, Combustion and Heat Transfer Processes In Internal Combustion Engines "Hungarian Journal of Industrial Chemistry Veszprem. Vol. 39 (1) pp.27-30 (2011).

[10]. V. Ganesan, "Internal Combustion Engines." Second edition, Tata Mcgraw Hill. ISBN 0-07-049457-6.

[11]. ANSYS Fluent User Manual.

[12]. Apex Innovations, VCR Engine User Manual.